International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 3, Issue 5, Dec 2013, 1-10

DEVELOPMENT OF INTEGRATED CNC-RP SYSTEM THROUGH CAD/CAM ENVIRONMENT

ABHISHEK AGRAWAL[1], RAJESH KUMAR SONI[2] & NEELESH DWIVEDI[3]

[1]Research Scholar, Department of Mechanical Engineering, University Institute of Technology, RGPV, Bhopal, India

[2]Department of Mechanical Engineering, People's University, Bhopal, India

[3]M.Tech Scholar, Department of Mechanical Engineering, Sagar Institute of Research & Technology, Bhopal, India

ABSTRACT

This paper propose to integrate a CNC Machining & Rapid Prototyping in a single system through a CAD/CAM interface. CNC Machining integrate with Rapid Prototyping is a method that has been developed which enables automatic generation of process plans from a geometric image of component. Using existing CAD/CAM software, which is well understood and normally used in machining operations the software developed manipulates the machining code from the ALPHACAM package and generates code and instructions suitable for RP technology. Two user-friendly programs were developed using Visual Basic to generate control code for a wax droplet RP system previously developed. CNC/RP follows the traditional method in RP and generates control instructions for the system from a STL file. The CNC/RP software is designed to generate control instructions for the system to build a prototype from CNC code, generated using ALPHACAM.

KEYWORDS: CNC Machining, CAD/CAM Environment, Rapid Prototyping, CAD Model

INTRODUCTION

Most commercial RP systems are based on additive processes whereby models are constructed by stacking cross sectional layers on top of one another. The additive RP systems are often limited in both geometric accuracy and material quality. Subtractive processes such as CNC machining have advantages over the limited choice of materials and the limited functionality of parts produced by additive processes. However, machining is not a completely automated method in either the process or fixture planning steps. There has been a need for a rapid machining system, but previous attempts to automate CNC machining have been approached from the perspective of traditional machining methods. It has become necessary to re-think how parts can be held, oriented, and then actually cut; perhaps borrowing methods from existing approaches to additive rapid prototyping processes.

Traditional machining requires extensive planning by a specialized and experienced machining technician. Moreover, the challenge of machining complex and intricately shaped components is daunting even on the most advanced machines. In traditional machining, the focus is typically on simpler geometries (holes, slots, planes, etc), or, when the geometry is more complex, as in an airplane wing, the shape is defined by known geometric functions. In these cases, a skilled machinist chooses cutting tools based on the desired shape (such as a drill bit for a round hole). With respect to this research effort, Rapid Prototyping and Manufacturing presents an enormous challenge in terms of geometric complexity and every part build is unique. In manufacturing, the time spent developing an optimal process plan for machining a component can be easily absorbed over a long production run. However, if truly one-of-a-kind components are required, then the engineering involved in planning and implementing the solution must be automated or eliminated altogether.

Process planning for CNC machining includes tasks such as determining: 1) how to clamp the stock material (fixture planning), 2) what orientation to approach the geometric surfaces with the cutter (setup planning), 3) what size and shape cutter to use (tool selection) and finally, 4) how to traverse the geometry with the cutting tool in order to create the surfaces (tool path planning). Computer Aided Process Planning (CAPP) systems have made some reduction in the time required to plan machining operations, but the time, skill and cost for one-of-a-kind machined parts is still dominated by the planning before machining and not the machining itself.

There are currently no approaches that are flexible enough to provide a universal solution for such various shapes as would be needed in rapid prototyping. To this end, CNC-RP is a new rapid machining process that has been developed to allow for push-button machining of complex parts with no advanced manufacturing skill required and no process planning time after the CAD model is obtained.

The CNC-RP process has been realized as a module in the ALPHACAM® package that automates virtually all of the process and setup planning tasks. Using several geometric algorithms and standardized tool and material libraries, this system can process a CAD model similar to the way an RP software interface processes an STL file. The interface computes setups, creates sacrificial support structures, generates tool paths and outputs the required NC code and setup instructions.

To process a part, the user simply loads the prescribed diameter and length stock material into the CNC machine and downloads the NC code for processing. Hence, CNC-RP has been implemented as one of the first truly push-button subtractive rapid prototyping systems, able to be implemented on a wide array of CNC milling machines.

INTRODUCTION OF USING CAD/CAM FOR RP

Many existing CAD/CAM software packages are used to generate the tool path for milling operations. There is a big similarity between the milling operation and deposition operation. The idea was to use existing CAD/CAM software to generate the tool path. Alpha CAM™ generates the tool path for CNC machining in G code format. To use this code for the PC23, software was developed called CNC/RP to process the G code by performing the next steps:

- Scanning the CNC code,
- Manipulating the tool path,
- Simulate the deposition of wax,
- Filtering the additional deposition in some areas,
- Translates the CNC code to X-language codes, which are the PC23 indexer instructions and
- Saves these instructions in the text file, or export the instructions to build the part.

Generate CNC Code for Rapid Prototyping

Alpha CAM is fully featured CAD/CAM software used to generate the tool paths for CNC machines it is developed by LICOM, which is part of the Planit group- a global company specialising in the development and distribution of software to the woodworking and engineering industries, situated in India. To generate a tool path suitable for use in RP the following must be considered:

- The part must be machined not the extra material. For example to produce the round rectangular in Figure by milling. The grey area must be machined but to build this in rapid prototyping the tool path for the white area has to be created.
- Wax droplet RP system has no need to move in Z direction; therefore the z value can be ignored for the operation.
- Parameters in the Alpha CAM software must be set according to the specification needed in the building operation, so the tool diameter can be set as a deposition wax diameter, and the milling depth as droplet height. A very small tool diameter can be set in Alpha CAM to generate a very precise tool path.
- All non-machining movements should be rapid, to enable the software to distinguish between the build movement and the movement without building.
- All the geometry in the same level has to be machined before the next level.

Example of Using Alpha CAM to Generate CNC Code for RP

This is an example of using Alpha cam to generate machining code for a rectangular pocket with a 21 mm length and 18mm in width to the depth of 6 mm, these are the Steps to generate a tool path in Alpha CAM

Step 1: Open the software:

Step 2: Define the material:

Step 3: 'Define the shape to be machined

Step 4: Tool selection

Step 5: Tool Directions

Step 6: Pocketing: Click on the “Pocketing” icon

Step 7: Pocketing

Step 8: Tool Path Resulted

Step 9: Save NC code for this process

Development of CNC/RP Software

CNC/RP user-friendly software is designed to generate control commands for the PC23, to build a prototype from CNC code, which was generated in Alpha CAM. When CNC/RP is executed the program opens the selected CNC file as a sequential input file, reads the text line-by-line, extracts all needed information such as G codes, the displacements in x, y, z and R, manipulates the tool path, simulates the deposition of wax, filtering the additional deposition in some areas, and then translates the CNC code to X-language codes, which are the PC23 indexer instructions to drive the motors.

The program flow is divided into seven steps:

- Extract the tool path.
- Drawing the CNC tool path.
- Simulate.

- Correction.
- Filtering.
- Build the instruction command file for PC23
- Save

CNC Alpha CAM text file can contain some comments, which may cause errors in identifying the tool path, so another procedure has to be taken to check for only the numerical values after the command strings. Is Numeric(a) function returns true if the whole string is numerical, this function is used to check only the digit after X,Y, Z, I, J and R. If Is Numeric (a) returns true, Val (str) function continues to read the string and returns the numerical values only. The program saves the numerical values only in a new text file using Write command. ALPHACAM can be also set to generate the code without any comments.

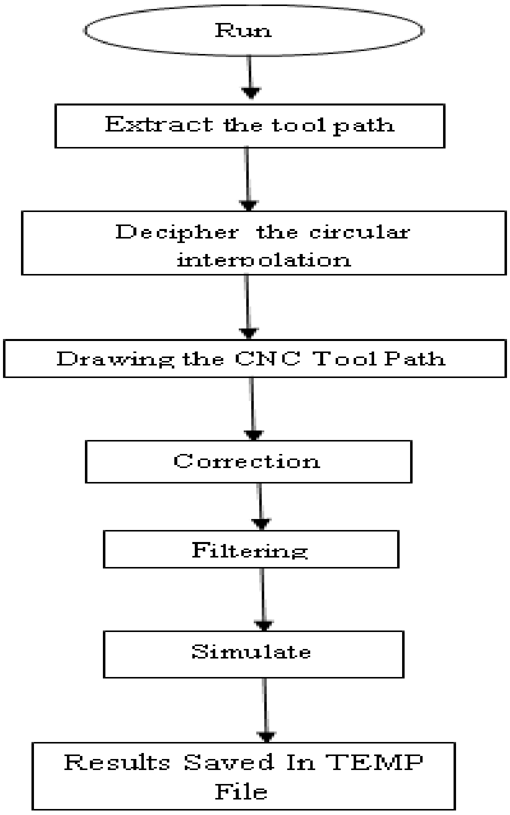

Figure 1: CNC/RP Run Algorithm Flow Chart

CNC-RP METHODOLOGY

This rapid machining process is based on a setup strategy whereby a rotary device is used to orient round stock material that is fixed between two opposing chucks. Rotating the stock using an indexer eliminates the inherent problem of retaining reference coordinates associated with re-clamping a part in a conventional fixture. For each orientation, all visible surfaces are machined and a set of sacrificial supports keep it connected to the uncut ends of the stock material. Once all operations are complete, the supports are severed (sawed or milled) in a final series of operations and the part is removed. The setup and steps to this process are illustrated in Figure 1. As an example in this figure, a component is being machined using sacrificial supports to retain the part at its ends along the axis of rotation. This method of using one axis of rotation for indexing between setups is obviously not capable of machining *all* parts of extremely complex shape. Parts with

severely undercut features or complex features on three or more mutually orthogonal faces may not be machinable with this approach. In particular, this setup strategy assumes that *some* axis of rotation exists such that all surfaces are visible.

The rapid machining process incrementally creates the part by machining layer by layer for each orientation, thus it is not very fast and would not be a good choice for production manufacturing. However, this process is extremely *rapid* because it can almost automatically begin processing a part directly from a CAD file. The research so far has developed algorithms that automatically analyze the part geometry and determine 1) if the part is machinable, 2) if so, how many orientations are required and 3) all the process parameters including the stock size, tool size, feeds, speeds, sacrificial support geometry and the sequence of operations This new rapid machining process is novel in that it approaches the planning steps in a “feature-free” manner using robust automated methods. That is, geometric algorithms calculate the steps necessary to machine all the surfaces of the object. The process begins with a 3D model and generates the required processing code for a computer numerically controlled (CNC) milling machine.

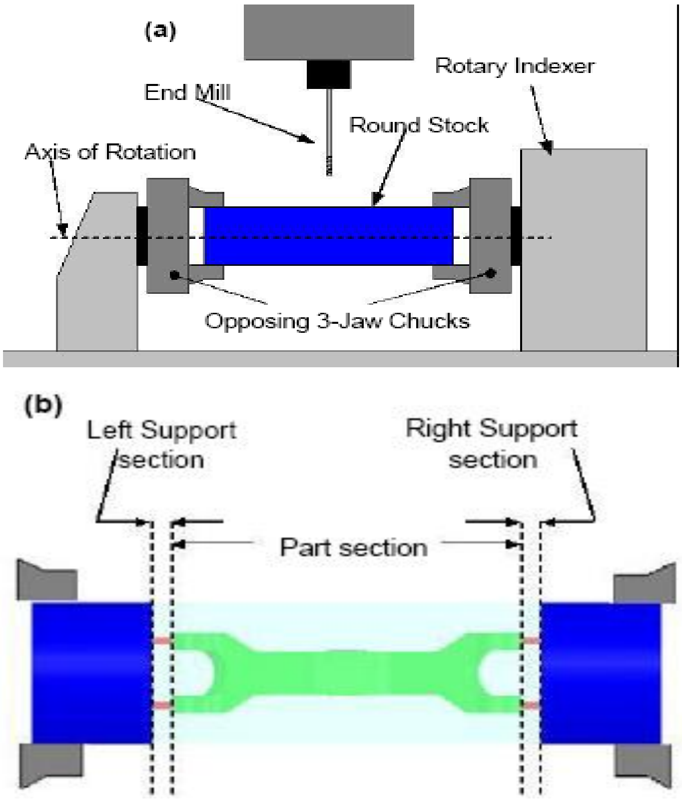

Figure 2: Rapid Machining; (a) Set up, (b) Sections Machining Approach

INTEGRATION OF CNC-RP IN CAD/CAM

The process is initiated after the CAD model of the part is loaded in the CAD/CAM system. Next, a sequence of steps is initiated including 1) analyzing the part for an axis of rotation 2) establishing a work coordinate system 3) attaching sacrificial supports 4) determining setup orientations about the axis, 4) generating a roughing process to remove the bulk of material and 5) generating a finishing process to create the surface geometry. Borrowing from traditional RP, this system does use the STL file format; however, not for creating the final NC code. In this research, we use the STL file for a feature-free analysis of the CAD model that enables us to determine all the process and fixture plans. Several geometric algorithms and heuristic tools have been developed to work with either the STL file or a slice file from the STL. However, the original native CAD file is preserved through the process. As such, the CAD model provides the drive surfaces for tool path planning, which eliminates the typical error associate with the STL format.

Depending on the step in the process, we export varying "quality" STL files, by changing the chordal deviation of the output. This is done because we can greatly reduce the computational load on the geometric algorithms, when precision is less important. There one path flow for the CAD model and then several "offline" analysis steps that are performed on STL files.

Axis of Rotation Planning

Since the CNC-RP process requires an axis of rotation, the part needs to be analyzed to determine if one exists, and if so, choose the best axis for the process. The axis of rotation can be initially searched based on surface visibility, a necessary condition for mach inability. For many parts that would be created with CNC-RP, these axes are most likely the orthogonal axes of the system coordinates.

That is, most designers work with a basic 3-view modeling approach, adding features to the views of the orthographic projections of the part (top-front-side sketching in most CAD systems). Similarly, one could argue that most parts intended for manufacturing have been designed for as few setups as possible. So, the first step in the analysis is to simply check the x-, y- and z-axes of the part and determine their visibility. In addition, the diameter of the part along each of these axes is found, since smaller diameter materials reduce cost and tooling requirements.

In order to implement this in CAD/CAM, the part model is exported as an STL file using a C program initiated by a custom toolbar. They STL model is then sliced along each of the 3 axes. The visibility analysis is completed for each of the three axes, along with a calculation of the diameter of the part along each axis.

Establishing a Coordinate System

The coordinate system for this process must be safely chosen to avoid any chance for tool collisions during processing. Thus, a simple translation of the part is made before processing, to shift the part to the negative x-space and centered on the axis of rotation. This step both ensures that the work coordinate system is consistent for each part and that the part is placed in the center of the stock material. The STL file exported in the previous step reveals the bounding box of the part, which is used to shift the model in the CAD/CAM space.

In addition, this approach can ensure collision free processing by establishing a safe working space between the chucks of the fixture system (Figure 3). As shown in the figure, the model can be placed in a location such that tool paths are contained in a safe region (tool collision cannot occur). Moreover, the user will not have to set any tool or work offsets before running a new part. In fact, since the length of the stock is defined in a setup sheet, the tailstock position is set by the length of the stock, therefore no additional checks are required during setup.

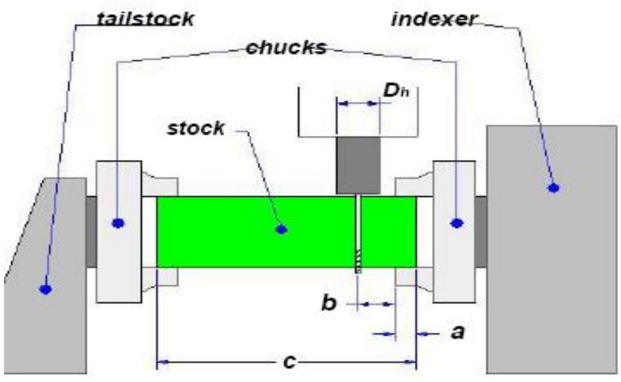

Figure 3: Fixture Setup and Parameters for Coordinate System Setting

Stock length: ***c*** = L_p *+ 2a + 2b where:*
$\boldsymbol{L_p}$ = Part length
a = Clamping length
b = Collision offset (*x*) = .5D_h +.5 D_{tMAX}
$\boldsymbol{D_h}$ = Diameter of tool holder
$\boldsymbol{D_{t\,MAX}}$ = Diameter of largest tool

Adding Sacrificial Supports

A significant challenge in using CNC machining to rapidly create custom shapes lies in the fixturing **[W1].** The concept of *flexible fixturing* has been a popular topic of previous research efforts; unfortunately a completely autonomous fixture design system has yet to be developed **[B1].** The difficult challenge in machining unique part geometries is that the fixtures that hold the material must be generated automatically. The proposed approach to fixturing for this research borrows from the general idea of *sacrificial supports*, which are used in existing RP systems (i.e. SLA). In this work the general design intent is retained; however, the physical requirements for the support structures are very different. The goal is to have a fixture support solution that is created in-process and is customized for each unique geometry. Specific to rapid machining, the fixture supports need to allow the part to be rotated about the axis while providing access to as much of the surface as possible (Not necessary in additive RP processes such as SLA). Conventional fixturing methods for CNC often utilize vices, clamps, vacuum surfaces, etc. These approaches occlude visibility to a significant amount of the part or make it difficult to reorient if multiple setups are required. Visibility is so important, simply because it is the *necessary* condition for being able to cut the surface with a machine tool and to finish cutting the part in as few setup operations as possible.

In the proposed method, the *sacrificial supports* are added to the ends of the CAD model automatically. In this manner, the part remains attached to the stock material throughout the set of machining operations. This leads to better geometric accuracy when we leave the object secured to the parent stock material rather than unclamping, removing, replacing and re-clamping it as in traditional machining setups. The details of the sacrificial support design methodology cannot be presented in this paper, but the approach can be summarized. In this approach, we consider a worst-case condition for the support design and drive the design based on a maximum allowable deflection.

This deflection is considered most significant in the form of twist angle (we approximate with a fixed-fixed statically indeterminate beam model). The design involves a layout of as few as *two* and as many as *four* supports in the layout. The design goals are to provide a layout with supports attached 1) near the ends of the part to avoid surface occlusion and 2) far from each other to provide a stiff structure. In order to ensure deflection is limited, we design the diameter for the *two* "permanent" supports, assuming there are in fact only *two* supports, and then add up to *two* more "temporary" supports if possible. As such, the first two supports are designed based on a simple concentric beam model using a maximum deflection input:

$$\theta_{max} = \frac{(D_t^2 R^4 + D_t L r_1^4)2T}{(D_t r_2^4 R^4 + L r_1^4 r_2^4 + D_t r_1^4 R^4)\pi G}$$

Where: θ_{max} is derived from a linear tolerance ($R\sin\theta_{max}$ =0.5 tol)

R is the radius of the stock material

$\boldsymbol{D_t}$ is the largest tool diameter (which determines the lengths of the support) $\boldsymbol{r_1}$ and $\boldsymbol{r_2}$ are the two support diameters

Orientation and Tool Path Planning

Once the sacrificial supports are added to the CAD model, they become part of the drive surfaces for the machining process. The next step is to determine the set of orientations for *Rough* and *Finish* machining of the part. In addition, the depths of cut, step down, feeds and speeds must be determined for each orientation. Once again, the model is analyzed for visibility about the axis of calculations using cross sectional geometry rotation. The slice file generated along the axis of rotation yields a set of data that can be used to not only determine part geometry and depths of cut, but to establish the setup orientations.

CONCLUSIONS

The implementation in a CAD/CAM environment enables CNC -RP as an effective rapid prototyping method for use on existing CNC machines. As such, the process can be used for short run production, custom manufacturing and prototyping for certain applications. We have developed many algorithms to process STL file, slicing algorithm, contour sorting algorithm and contour filling algorithm. The CAD model data can be completely transferred to CAM program for producing the product. We have also developed translating algorithm for CNC code, which provides the capability to use G code from machining packages such as Alpha CAM and Pro Engineering etc for RP. In addition, the same CNC machine can also be free to create production parts in conventional applications. The impact is that there is no need for a specialized RP machine for these functional prototypes and parts. The system is currently being tested as a method for creating spare parts for legacy equipment in the agricultural industry. In contrast, however, CNC-RP is generally more capable in surface finish, since layer depths can be controlled to very small values, as compared to additive RP processes. The greatest advantage however, is that CNC RP has exceedingly better capability in using a variety of materials to create truly functional parts.

REFERENCES

1. **B1.** Bi, Z.M. and Zhang, W.J., “Flexible fixture design and automation: Review, issues and future directions”, International Journal of Production Research, Vol. 39, No.13, pp. 2867-2894, September 2001

2. **C1.** Chen, Y. H., Lee, Y.S., and Fang, S. C., “Optimal cutter selection and machining plan determination for process planning and NC machining of complex surfaces”, Journal of Manufacturing Systems, Vol. 17, No. 5, pp. 371-388, 1998

3. **F1.** Frank, M.C., Wysk, R.A., and Joshi, S.B., “Determining Setup Orientations from the Visibility of Slice Geometry for Rapid CNC Machining”, Journal of Manufacturing Science and Engineering, Transactions of the ASME, Vol. 128, No. 1, pp. 228-238, 2006

4. **F2.** Frank, M.C., Wysk, R.A., and Joshi, S.B., “Rapid Planning for CNC Machining – A New Approach to Rapid Prototyping”, Journal of Manufacturing Systems, SME, Volume 23, No. 3, pp. 242-255, 2004.

5. **H1.** Hassold, R., “CNC machining as a rapid prototyping technique”, Modern Machine Shop, Vol. 68, No. 5, pp. 68-73, October, 1995

6. **J1.** Joneja, A., and Chang, T.C., “Setup and fixture planning in automated process planning systems”, IIE Transactions, Vol. 31, No. 7, pp. 653-665, 1999

7. **L1.** LI, Y. and Frank, M.C., “Computing Non-Visibility of Convex Polygonal Facets on the Surface of a Polyhedral CAD Model”, Computer Aided Design, Vol. 39, No. 9, pp. 732-744, 2007

8. **L2.** LI, Y. and Frank, M.C., "Machinability Analysis for 3-axis Flat End Milling", Journal of Manufacturing Science and Engineering, Transactions of the ASME, Vol. 128, No. 2, pp. 454-464, 2006.

9. **M1.** Maropoulos, P.G., "Review of research in tooling technology, process modeling and Process planning, Part 2: Process Planning", Computer Integrated Manufacturing Systems, Vol. 8, No. 1, pp. 13-20, 1995

10. **S1.** Schmidt, Joseph. W., "CNC machining - the Other Rapid Prototyping Technology", SAE Special Publications, Vol. 1233, Automotive Concurrent/Simultaneous, Engineering, pp. 89-91, 1997

11. **W1.** Wang, F.C., Marchetti, L. and Wright, P.K., "Rapid Prototyping Using Machining", SME Technical Paper, PE99-118, 1999.